

Study Group 20

ELECTRONIC FUNDAMENTALS



Service Practices 39:

PRINTED CIRCUITS

Service Practices 40:

PRACTICAL TELEVISION ANTENNAS
AND TRANSMISSION LINES



RCA INSTITUTES, INC.

A SERVICE OF RADIO CORPORATION OF AMERICA

New York,

N. Y.

ELECTRONIC FUNDAMENTALS

SERVICE PRACTICES 39

PRINTED CIRCUITS

- 39-1. Historical Background**
- 39-2. Manufacturing Printed Circuits**
- 39-3. Service Techniques**
- 39-4. Making Your Own Printed
Circuits**



RCA INSTITUTES, INC.
A SERVICE OF RADIO CORPORATION OF AMERICA
HOME STUDY SCHOOL
350 West 4th Street, New York 14, N. Y.

Service Practices 39

39-1. HISTORICAL BACKGROUND

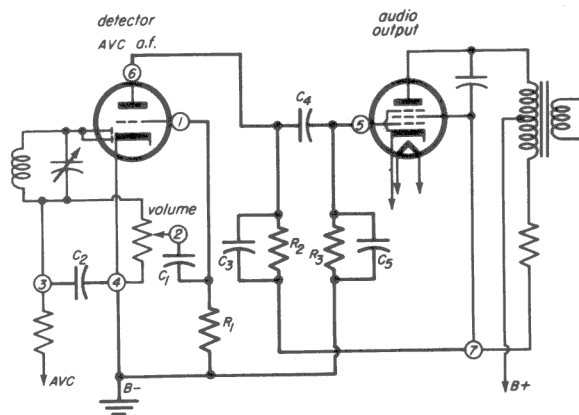
Printed circuits were developed as an answer to certain critical needs of the electronics industry. The history of their development is in itself a fascinating story of how the electronics industry met a challenge.

At the beginning of World War II, the electronics industry was called upon to produce an unprecedented amount of equipment at record speed. Even before the United States entered the war, factories were called upon to produce material for shipment to our allies. But this enormous demand created certain problems, notably a shortage of skilled electronics technicians to make, test, and repair the equipment, and the need to conserve raw materials that were already in short supply. Furthermore, equipment made for military use had to withstand more violent conditions and occupy less space than equipment made for civilian consumption.

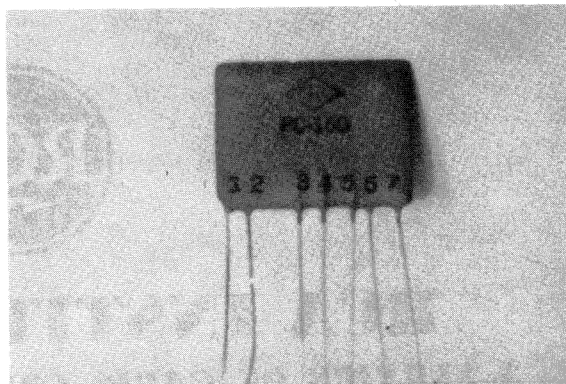
One of the solutions offered to the problems of the electronic needs of the armed forces was the printed circuit. As early as 1940, the Erie Corporation used printed wiring techniques. The Bureau of Standards and the Centralab Division of the Globe Union Company also pioneered in the field. One of the noteworthy contributions of the Globe Company was the use of printed circuits in proximity fuses, which are electronic circuits used to cause a bomb to detonate before it strikes its target. In 1943, RCA was building proximity fuses, employing printed circuit boards. Such circuits had to withstand the tremendous acceleration produced when a bomb falls, and they had to be shaken loose, and the fact that they are practically two dimensional, made them ideal for this application.

The Bureau of Standards worked on various technical problems connected with the manufacture of printed circuits in the attempt to improve printed circuit techniques.

Stimulated by wartime discoveries, a considerable expansion of printed circuit manufacture occurred after the war. For example, Centralab, working with principles evolved during the manufacture of proximity fuses, developed *couplates*, a name coined to describe a printed circuit that includes, within a ceramic envelope, all the components needed to connect two resistance-



(a)



(b)

Fig. 39-1

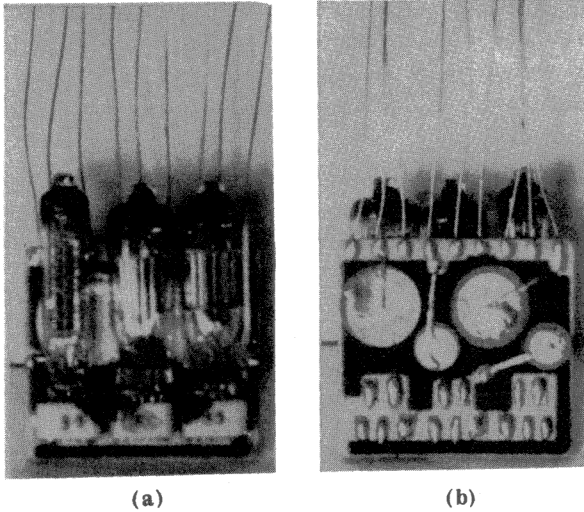


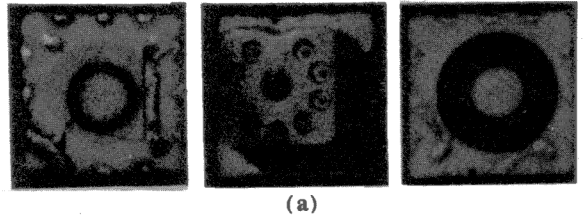
Fig. 39-2

coupled audio-amplifier tubes. Sometimes, these units are called *coupling plates*. This type of printed circuit can include printed capacitors, printed resistors, and printed wiring. A large variety of coupling plates can be found in contemporary portable radios, electronic organs, television receivers, radiation detectors, and other electronic apparatus. A schematic diagram of a *couplate* used between two audio stages is shown by the heavy lines in Fig. 39-1a, and the actual couplate itself is shown in Fig. 39-1b. A front view of a three-stage amplifier unit that uses printed wiring is shown in Fig. 39-2a and a rear view of the same unit is shown in Fig. 39-2b. By means of using couplates and other printed circuits, entire circuits can be wired into an electronic device with hardly any more difficulty than was required to wire a single component into a conventional circuit.

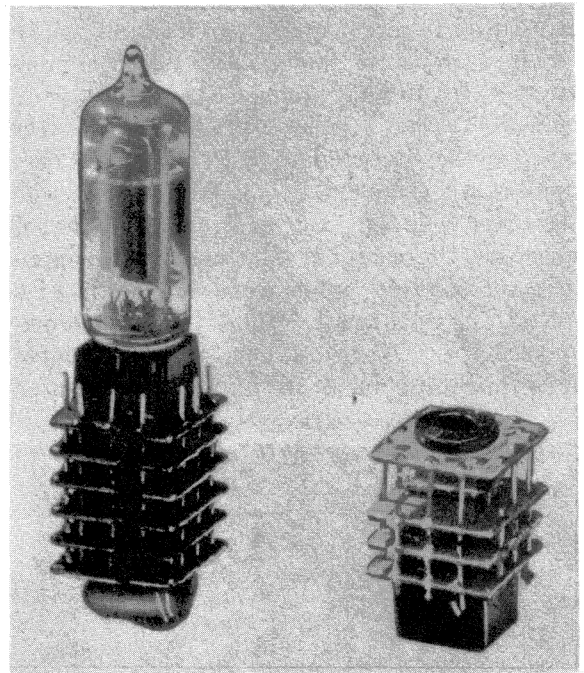
Another type of circuit that utilizes printed-circuit techniques is called a *module*. Modules were developed as the result of a project undertaken by the Bureau of Standards for a U.S. Navy project known as *Tinkertoy*. A module is an assembly of components that together make up a larger component or circuit. As shown in Fig. 39-3a, wiring and components can be printed on a separate ceramic wafers. These wafers can be stacked up, one above the other, as shown in Fig. 39-3b. The electrical connections between wafers are provided by

some of the *riser* wires, which are the vertical wires that connect the wafers. The riser wires fit into notches on the edges of the wafers. Some of the riser wires are not electrical connections. They simply provide physical support for the module.

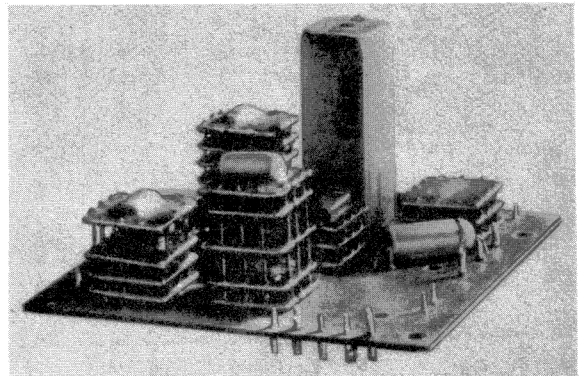
Modules are often used by design engineers. The riser wires of a module can be plugged directly into a printed circuit



(a)



(b)



(c)

Fig. 39-3

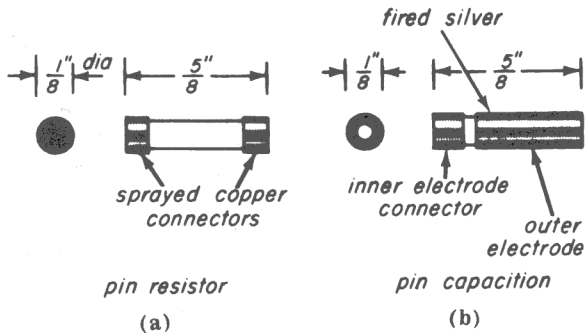


Fig. 39-4

board, as shown in Fig. 39-3c, and can be removed just as easily. Contrast this with the time and labor needed to wire up a special circuit whenever one is needed.

Another form of modular assembly is the *Packaged Assembly Circuit* (P.A.C.) developed by the Erie Corporation. The modules in this design consist of components in tubular form (Fig. 39-4). Any components that can be made in this tubular form can be included in a P.A.C. The components are attached by machine to clips on a specially designed phenolic plastic base, and soldered in place. The base has printed wiring. Figure 39-5a shows a schematic of a multivibrator circuit. The tubular components for this circuit are shown mounted on a phenolic base in b, and the printed

wiring that connects the tubular components together on the opposite side of the base, is shown in c. Several P.A.C. units can be installed on a printed circuit base, as shown in Fig. 39-6. P.A.C. units are easy to replace.

A module developed by the Eastman Kodak Company consists of standard components mounted between a pair of circuit boards that have printed wiring on them (Fig. 39-7). The name of this modular assembly is the *Paraplate Bundle*. It was developed primarily to solve the problems of low-volume machine production and frequent change of product at reasonably low cost.

A great deal of development work is being carried out to further simplify and standardize printed circuitry. One firm is in the process of developing air-conditioning controls incorporating printed wiring. Electronic organs are typical of the complex, repetitive devices where printed circuits can afford great savings in labor, along with other advantages. Figure 39-8 shows a printed circuit board used in a Westinghouse welding control unit. In a, the front view, with components mounted, is shown. In b, the rear view, you can see the printed wiring

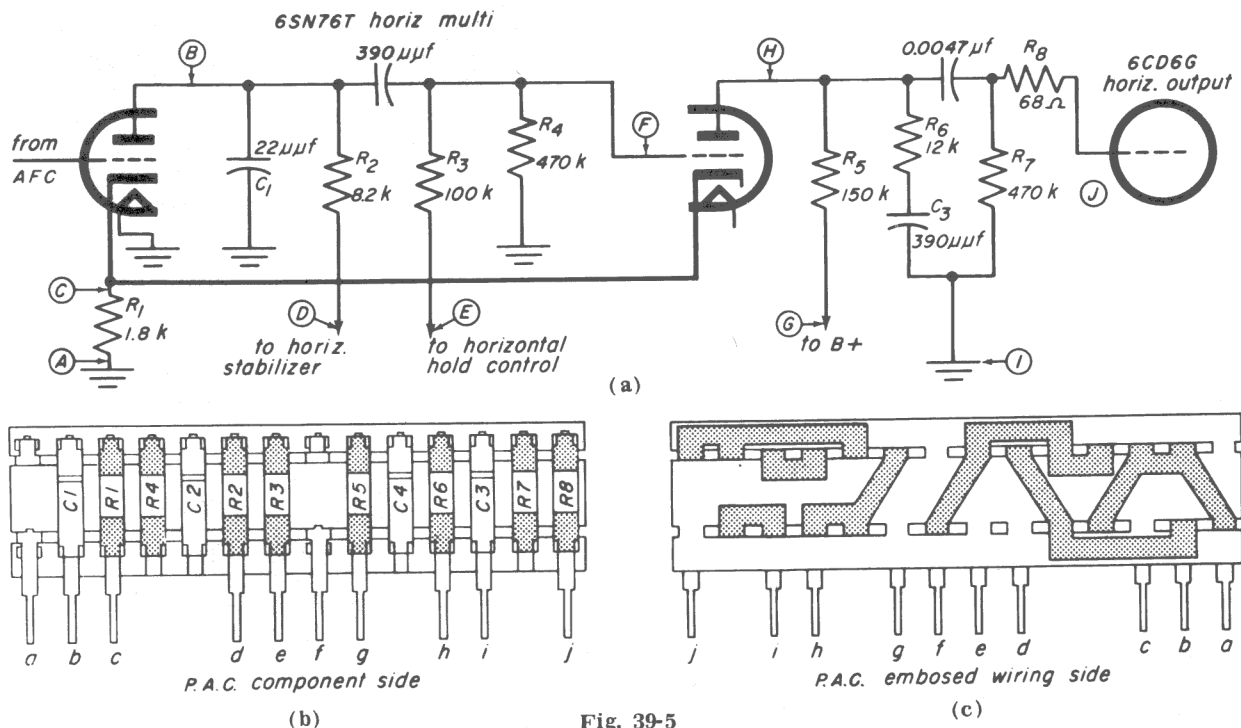


Fig. 39-5

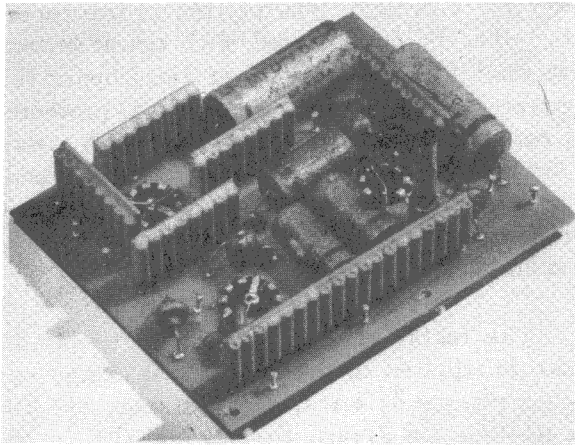


Fig. 39-6

that connects the components. Figure 39-9 shows printed wiring used in a Stromberg-Carlson Carillon (electronic-mechanical substitute for bells). A carillon is a set of bells used for playing melodies. The Stromberg-Carlson carillon produces the same effect by electronic means. At least one firm is trying to develop printed circuitry in which most of the base material on which the circuit is printed is also removed in order to achieve certain performance characteristics. With transistors coming into general use, printed circuits used with them will take on characteristics required by low current values and extreme compactness. Possibly transistors will be designed for inclusion in modular assemblies. Barrier-type transistors may eventually be printed in the way that wiring and other components are printed. Sub-miniature components can make good use of printed wiring. Movable or adjustable components, such as switches

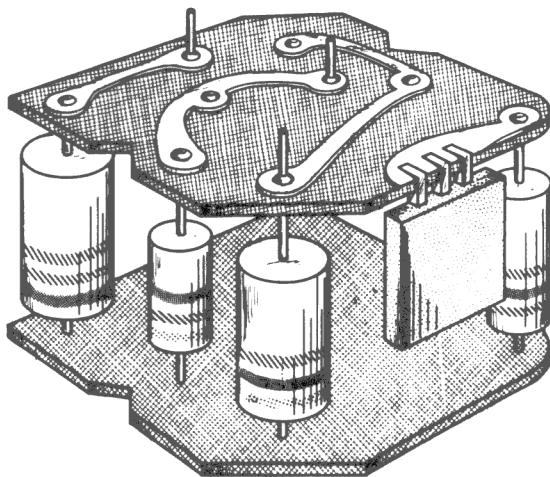
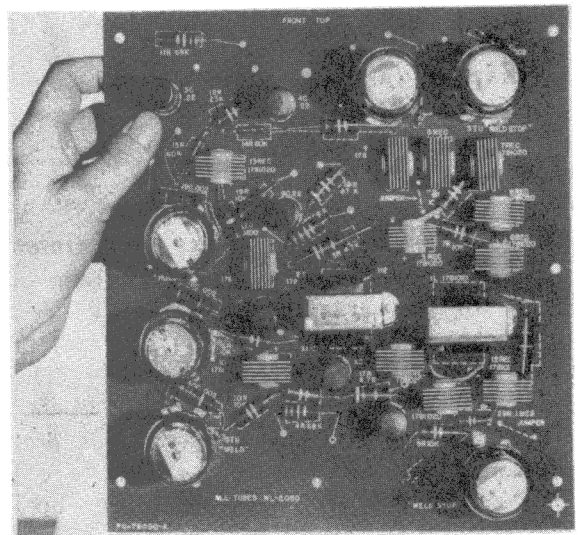


Fig. 39-7

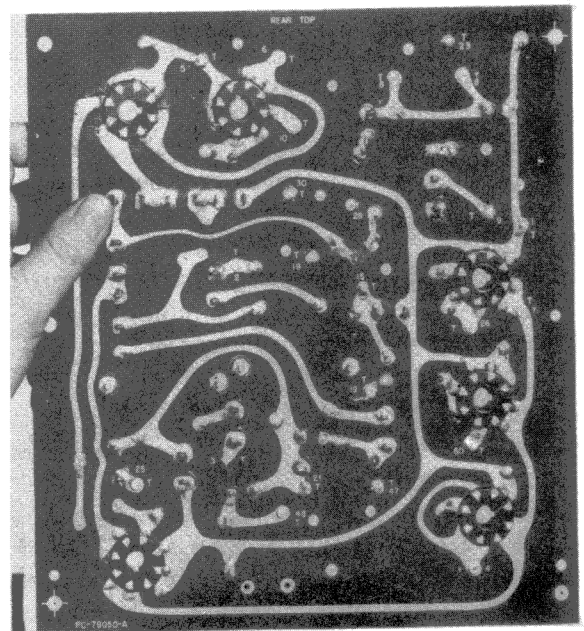
and potentiometers will probably be directly included in the printed circuit more often. A recently invented radiation detector could be printed on a high-quality insulating base. Lead sulfide or selenium photocells probably will become printable to be directly included in the circuit.

39-2. MANUFACTURING PRINTED CIRCUITS

In order to understand printed circuits, it is useful to know how they are made. The Bureau of Standards list twenty-six separate



(a)



(b)

Fig. 39-8

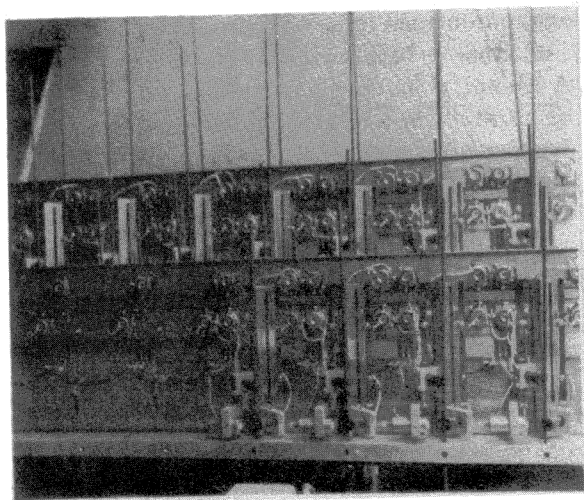


Fig. 39-9

and distinct (although sometimes related) methods of producing printed circuits. We will consider the methods most commonly used.

Base. Most printed circuits are printed on a nonconducting base made either of plastic or of ceramic. A frequently used type of base is made of laminated paper or plastic impregnated with phenolic. The nonconducting base material is covered with a sheet of metal foil, which is held to the base by an adhesive. The choice of the base material used depends upon both mechanical and electrical considerations. For instance, if a printed circuit has to withstand severe mechanical strain or high temperatures, the base must be made of a material with the appropriate characteristics. Also, the board should have good *dimensional stability*; that is, it should keep its original dimensions under various circumstances so that the shape and spacing of the printed circuit will not change. Among the electrical considerations are dielectric strength, power factor, insulation resistance, and loss characteristics.

Etching. Etching is the method most commonly used for printing circuits. In general, etching involves using acid to cut a pattern into a metal base. When printed circuits are etched, the base usually consists of an insulating material covered with copper foil. The circuit that is to be printed is outlined on the copper in *resist* enamel, an enamel that does not dissolve in acid. Then the entire copper-foil covered base is

washed with acid. The portion of the copper foil that is not covered with resist enamel washes away. The part of the copper foil covered with resist enamel is protected from the acid and does not wash away. Among the etching solutions are chromic acid, ferric chloride, nitric acid and a combination of dilute nitric acid and sulphuric acid.

The resist enamel outline of the circuit can be applied to the copper foil in any one of a number of ways. For instance, the circuit can be drawn directly on the copper with a brush dipped in resist enamel. A stencil with the outline of the circuit to be printed can be held against the foil covered base and the resist enamel can be either painted or sprayed on the stencil. The resist enamel goes through the lines cut in the stencil, so the copper foil on the insulating base has a protective coating of resist enamel in the same pattern as the lines cut in the stencil.

A widely used process of applying resist enamel is the photographic process. Let's go over this process step by step. A cross section of the base on which the circuit is to be printed is shown in Fig. 39-10a. The circuit to be reproduced is carefully drawn, and a photograph is taken of the drawing. The negative of the photograph (Fig. 39-10b) is transparent where there were lines on the drawing and opaque where there were no lines. The copper-foil covered base is coated with a special resist enamel (Fig. 39-10c). The negative is placed over the resist enamel (Fig. 39-10d). This resist enamel becomes insoluble in acid only if it has been exposed to light. Light from a carbon-arc lamp is shined on the negative. Only where there are circuit lines does a significant amount of light pass through the negative. Therefore, the resist enamel is hardened by light only where there are circuit lines. The rest of the resist does not harden. Then the negative is removed and the base is immersed in a developer liquid. This liquid further hardens the resist that has been exposed to light and dissolves the unexposed resist. As a result, when the base is washed with acid (Fig. 39-10e) the copper is protected from the action of the acid only by the hardened

resist. The unhardened resist and the copper foil under it are washed away (Fig. 39-10f), leaving the circuit printed in copper (Fig. 39-10g).

In another method using the photographic process, the copper foil is coated with a light-sensitive resist that becomes *soluble* in certain solutions after it has been ex-

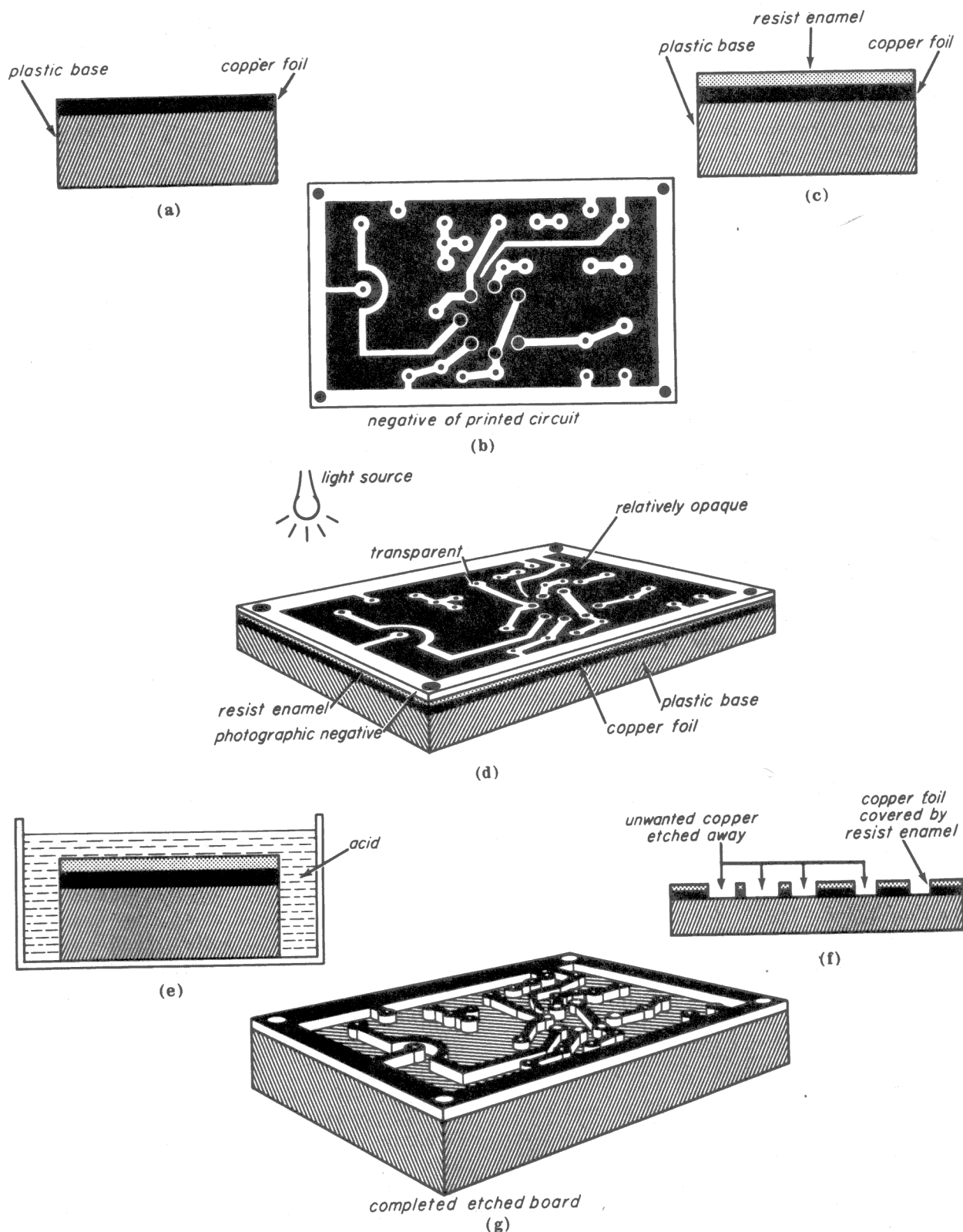


Fig. 39-10

posed to light. A *positive* print of the circuit pattern is placed on the resist coated board. The board is exposed to light and then developed in the solution which washes away *only* those portions of the resist exposed to light, leaving the circuit pattern protected against the etchant. This type of process does not need a hardener solution; washing in water and drying is sufficient.

Another process that is used to apply resist to the copper-foil covered base is called the silk-screen process. A piece of silk of very fine mesh has the pattern of the circuit transferred to it photographically. Then the piece of silk is stretched tightly against the copper-foil covered base. The silk screen acts as a stencil. Resist enamel is applied to the silk screen. The resist goes through the screen and coats the copper foil with a pattern that corresponds to the circuit.

Sometimes *decals* are used to apply printed circuits or resist. Decals are thin sheets of specially prepared cloth, paper, or plastic with a design printed on them. When the decal is placed against another surface, with the printed design against that surface, and a solvent is applied to the decal, the design is transferred from the decal to the other surface. (You may be familiar with the decals used to decorate Easter eggs.) A circuit pattern can be printed in resist enamel on a very thin transparent film, usually dried lacquer. This pattern can be transferred to foil. When the decal is washed with an appropriate solvent, the dried lacquer base is washed away; the resist pattern remains intact. Another method used is to make decals with silver conducting paint. These decals are then applied to a phenolic or ceramic base directly, and the silver circuit is transferred to the base. Curing the base with the silver on it in an oven makes the silver circuit pattern continuous and conducting.

Another way of applying the printed circuit to the base is by means of printing the circuit on the copper-foil in greasy ink. The ink acts as a resist, as shown in Fig. 39-11, which contrasts the photoetch process with the greasy ink process.

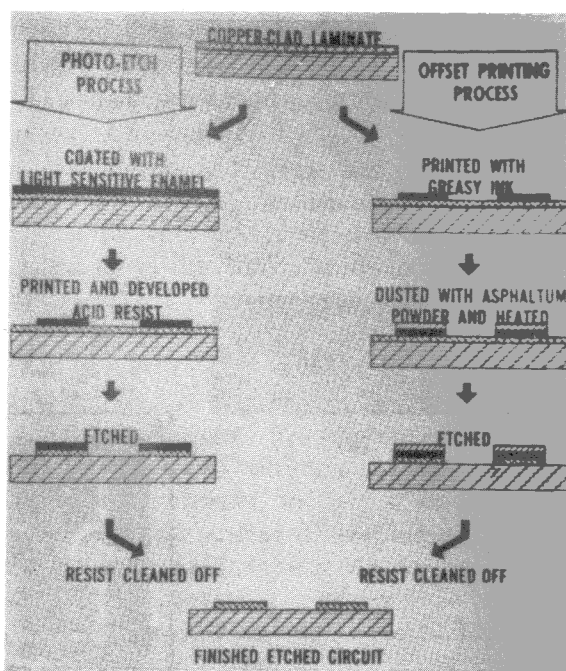


Fig. 39-11

The ink can be applied to many copper-covered bases in a short time, by means of a printing press, as shown in the simplified drawing of Fig. 39-12. After the boards are printed with the resist ink, they are etched.

Etching of printed circuit boards is most often done in multiples of the circuit, each piece passing through the process carrying a number of the final circuit layouts. These pieces are later cut into individual circuits.

Embossing. To emboss means to print a raised pattern. There are two widely used methods of embossing. One uses a die shaped in the form of the circuit to press the conducting layer into the plastic backing material, after which the remaining projecting material is milled off in a surface mill. In the other method, sometimes known as die stamping, a die of the circuit lines with sharp edges is pressed against the foil-covered base. The die delineates the circuit in foil as it is pressed down and attaches the metal to the base. Embossing of this type is done either with or without bonding material. The conducting material that is embossed on the base may be either copper or silver. Silver is a better conductor, less subject to corrosion and more ductile than copper. It can be used in em-

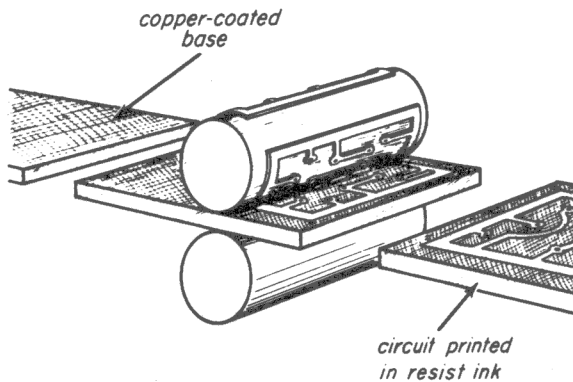


Fig. 39-12

bossed circuits because any silver not used can be recovered. Silver is rarely used in the etching process because the wasted metal is difficult to retrieve from the etchant. The chief disadvantage of the embossing methods is the cost of the dies. Therefore, embossing is suitable only for large scale production.

Metal-Spraying and Electroplating. In this method, hot metal is sprayed against the plastic base material. No bonding other than the adhesion of the hot metal is necessary. In some cases, the spray is applied through a stencil and delineates the circuit.

In a similar method, the entire base is coated with a bonding material. Then a very fine layer of silver is sprayed on to the base. Then resist is applied to all portions of the base that are *not* to be printed. As a result, the circuit pattern is surrounded by a resist and the circuit itself is printed in silver that is not covered with resist. The next step is electroplating. The thin silver layer serves as the plating conductor, and the copper or silver that is to form the printed circuit is built up on those sections of the board not coated with resist. When the plating is completed to the desired thickness, the resist, the silver layer under it, and the bonding material are removed with a solvent. The entire printed circuit board is thoroughly cleaned and coated with a solder resist, which prevents bridging of adjacent circuit lines by the solder. This method is very economical in its use of metal.

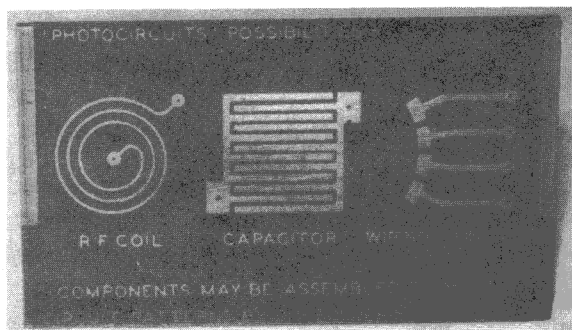
In another plating method, the circuit is delineated with hot sprayed metal. Then the metal of the circuit is built up by means of electroplating.

Fused Powder. The fused powder method is used in the manufacture of couplers, Ampec amplifiers, modules for the ACF and Erie P.A.C. systems, and the early proximity fuse printed circuits. The circuit wiring may be outlined on the prepared ceramic base in adhesive material or no adhesive may be used. Silver paint is screened onto the adhesive or the bare base. The silver paint and the adhesive, if any, are fused. A reducing agent is used to remove the oxygen that forms during the fusing action.

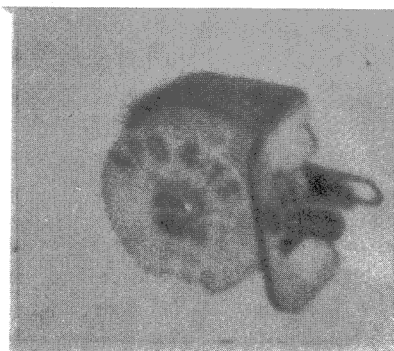
Resistors also are applied by the fused powder method. The resistance material, carbon or graphite of a definite composition, is mixed with a resinous binder, such as varnish, and screened onto the ceramic base. The resistance of the resistor depends upon the length and thickness of the deposit. The thickness of the deposit is controlled by the mesh of the screen and the viscosity of the mixture. The deposited resistive material, or *resistorfilm* as it is sometimes called, is then cured. It is in the curing process that the exact tolerances are obtained. In curing, the films are kept in an oven at fairly high temperatures for long periods of time. This stabilizes the electrical characteristics of the resistor film.

Plastic resistor tapes can also be applied to fused-powder printed circuits. The resistors are punched out to the exact length and width that corresponds to the desired resistance and wattage values. Then the bits of self-sticking tape are applied to the previously fused circuit wiring and cured in accordance with carefully prescribed directions.

In the Tinkertoy type of module, the fusing process can be used to attach certain types of components to printed wiring. The components used must be capable of withstanding the temperatures involved in the process. For example, silver-ceramic capacitors can be used.



(a)



(b)

Fig. 39-13

More frequently, components are attached to fused powder circuits by means of soldering. This method must be used when components will not withstand high temperatures, as is the case with most standard components.

Molding and Die-Casting. In the molding process, the desired circuit is cut into the base plate. The metal that is to form the printed circuit may be sprayed into the grooves; or the grooves may be filled with metallic powder and flash fired so that the powder will fuse.

In the die-casting method, the grooves are simply filled with cast metal, which is then machined or ground off for finishing. The latter method requires a base capable of withstanding the high temperatures involved.

Components. The majority of components used with printed circuits are standard components. However, there are some that are printed as shown in Fig. 39-13a, and others designed for use with printed wiring, as is the trimmer capacitor shown in Fig.

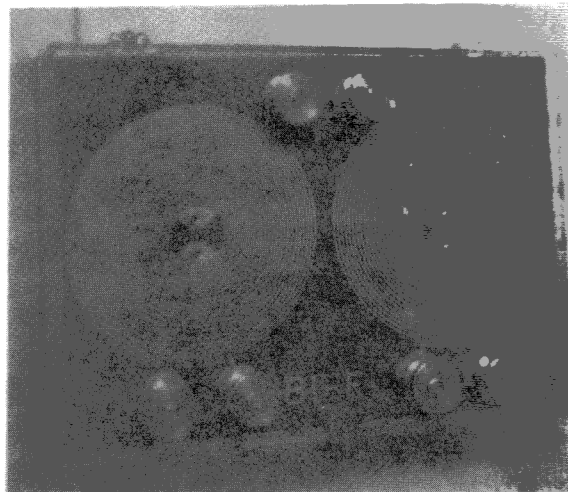
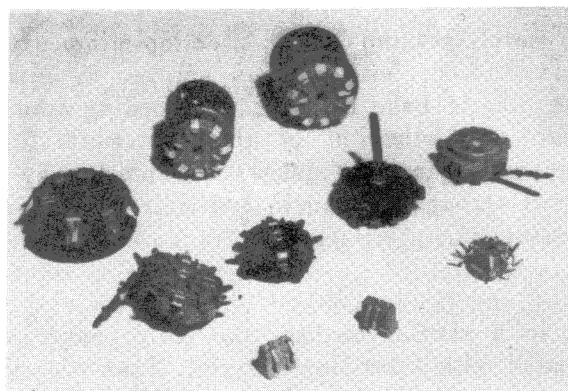
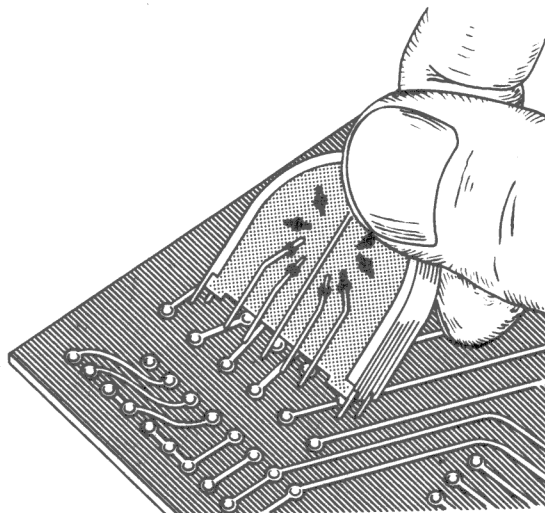


Fig. 39-14

39-13b. Sometimes, two or more printed components are supplied as a single unit, as in Fig. 39-14, which shows a printed circuit TV coupler used to couple two television receivers to the same antenna while

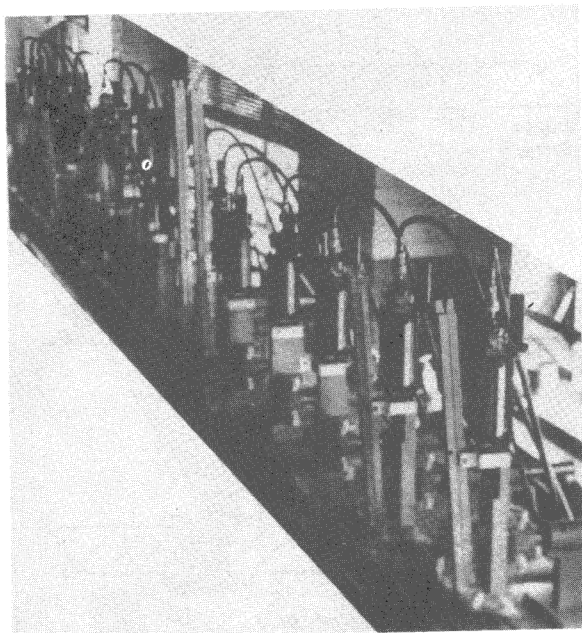


(a)



(b)

Fig. 39-15

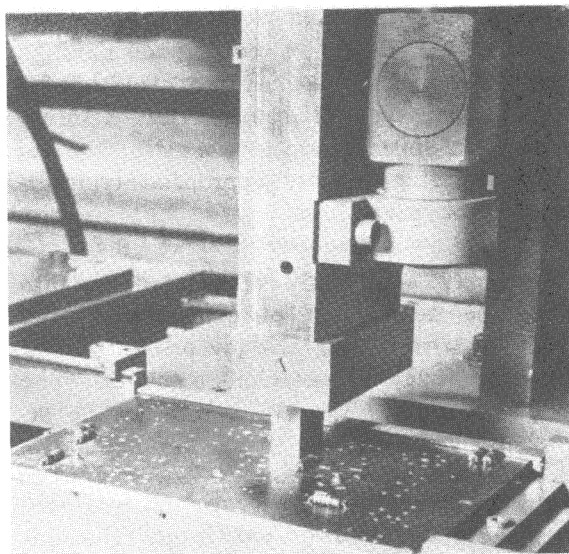


(a)

maintaining the 300-ohm line match required for efficient reception. Similar couplers for three and more receivers are available. Other frequently encountered printed components are i-f strips, loop antennas, r-f chokes, commutator and switch contacts, capacitors, and even strip-type waveguides.

Although printed resistors are used in mass production on special types of printed circuitry such as modules and couplers, they have not yet come into general use for the larger circuit boards. One of the reasons is that printed resistors are difficult to replace. Also, tape-type resistors require special curing. Recent developments in printed components include flexible printed cables, which are multiconductor vinyl-copper foil sandwiches made by the embossing process. The Remington Arms Corp. in 1948 produced printed vitreous enamel capacitors by means of building up alternate layers of vitreous enamel and silver paint and oven-curing them. Almost any desired capacitance or configuration of capacitors can be built up in this manner. These vitreous-enamel dielectric capacitors were built to military specifications to circumvent a shortage of mica and are used in some military equipment.

Many standard components have been modified for use with printed circuits. There



(b)



(c)

Fig. 39-16

are specially designed electrolytic and paper capacitors with special terminals, selenium rectifiers, special tube sockets, potentiometers, variable capacitors, trimmers, i-f transformers, etc. Figure 39-15a shows some special tube sockets designed for use with printed circuits in some industrial equipment, and Fig. 39-15b shows a right-angle tube socket. Using right-angle sockets makes it possible to keep the tubes parallel to the circuit board, thereby making the unit

in which they are used more compact. This type of socket results in great space savings when stacked-subassembly construction is used. In that type of construction, printed-circuit boards carrying a specific type of circuit are plugged into appropriately interconnected special plugs.

Assembly. Assembly of printed circuits means the attachment of components to a baseboard with wiring already printed on it. Components can be attached manually. The leads and lugs of the components to be attached are bent and cut to the desired shape and size and inserted into holes that have been provided for them.

In machine assembly, the machines in one motion trim and bend the leads and insert the component. Because of the force used for machine insertion, components must have sturdy terminals. At each machine station, the components are tumbled and guided to be stacked in the exact configuration required for insertion. One machine station can insert either one component or several components at once. The assembly system can be set up so that misalignment will cause the line to reject the part and the circuit board it was to be inserted in and provide an audio-visual signal for the operator to remedy the situation. An automatic assembly line at the Admiral factory is shown in Fig. 39-16. The automatic assembly system shown in Fig. 39-16a assembles the components to the board shown in Fig. 39-16b. The machines used are called *Autofab* and were developed by IBM. In order to prevent misalignment, components are being made with either nonsymmetrical terminals or special terminals, thereby making it impossible to insert the component in the wrong position. The product of the assembly line is shown in Fig. 39-16c.

Sometimes, eyelets are used in all component holes in order to make it easier for the machine doing the assembly to locate the holes with the end of the component leads. Figure 39-17 shows a cross section sketch of eyelet installation. In Fig. 39-18a, you can see one side of a board from an amplifier kit that uses eyelets for all component connections and for feed-through from the foils

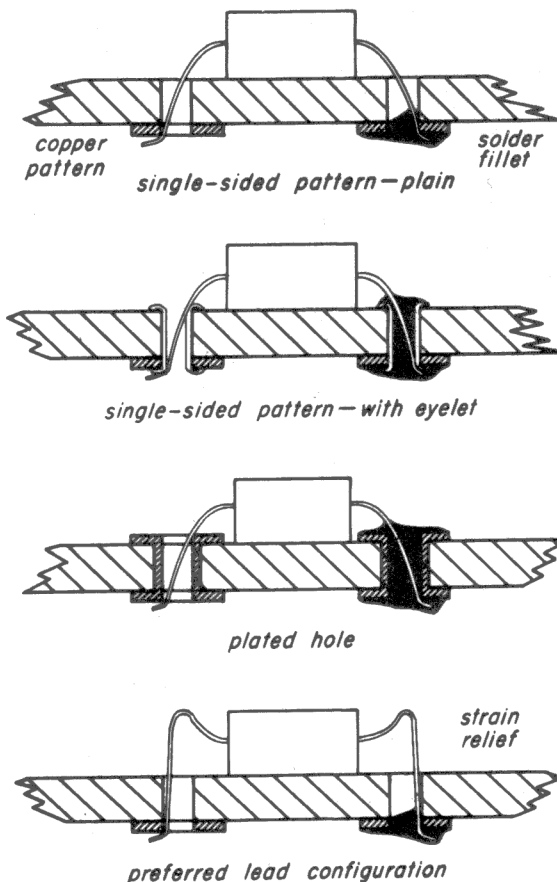
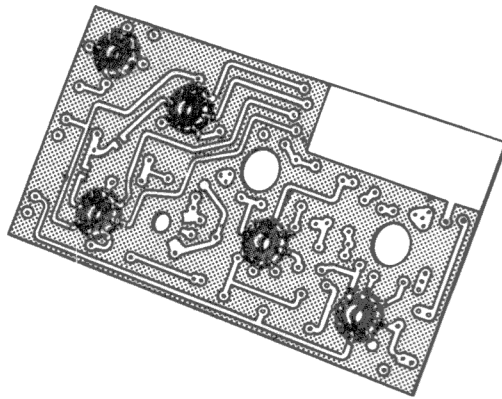


Fig. 39-17

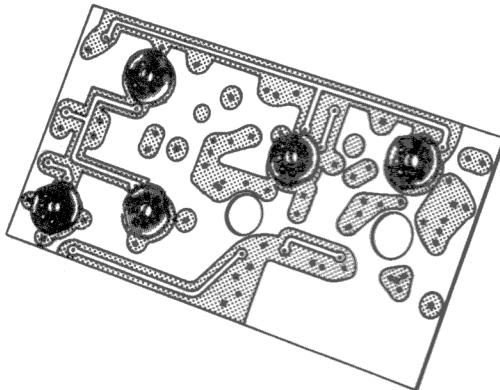
on either side, and in Fig. 39-18b, you can see the other side of the same board. However, eyelets have several disadvantages. If the eyelet is not snugged down on the copper conductor properly, trapped air may prevent proper soldering in the dipping process. *Swaging* the conductors on both sides of the board before soldering is sometimes done. Swaging is forming the leads with much pressure on both sides of the printed base in such a way that the leads press against both sides of the printed base very firmly. This also gives the leads a slope that can hold the components more firmly to the base.

With components mounted flat and snug on the board, no great need for the eyelets exists, provided the terminals are properly shaped.

After components have been assembled to the bases, they are soldered. One way of soldering is to solder manually, one joint at a time. It is important not to use too much



(a)



(b)

Fig. 39-18

heat, which will damage the printed circuit board. Small pencil-type irons ranging from 35 to 50 watts are recommended. To insure against too much heat reaching the foil, only the lead is heated. The solder is made to flow from the leads onto the foil conductor. The choice of the flux is extremely important; corrosive fluxes are always to be avoided. Too much flux can attract dirt and dust to the joints, which could then bridge the space between conductors and cause partial shorts. Cleanliness of the copper conductors and the component leads is essential. Hot dip pre-tinning of the circuit pattern is sometimes used to facilitate soldering. Silver plating is another means to the same end.

The method of soldering widely used in large-scale production runs, is called dip-soldering. In this method, all the solder joints are soldered at once. The printed circuit is held between the clamps of the dip-soldering machine (Fig. 39-19a and b)

and lowered into a bath of molten solder (Fig. 39-19c).

39-3. SERVICE TECHNIQUES

Techniques employed in servicing printed circuit boards differ somewhat from techniques that have been used in the past. However, experience has shown that printed circuit boards can be serviced easily when the proper technique and tools are used. Many technical publications have been published providing the technician with the necessary information required to enable him to effectively service printed circuit boards in an easy and simple manner. However, it is up to the technician to apply himself and learn to use the proper tools and develop the new techniques if he intends to keep up with progress in industry.

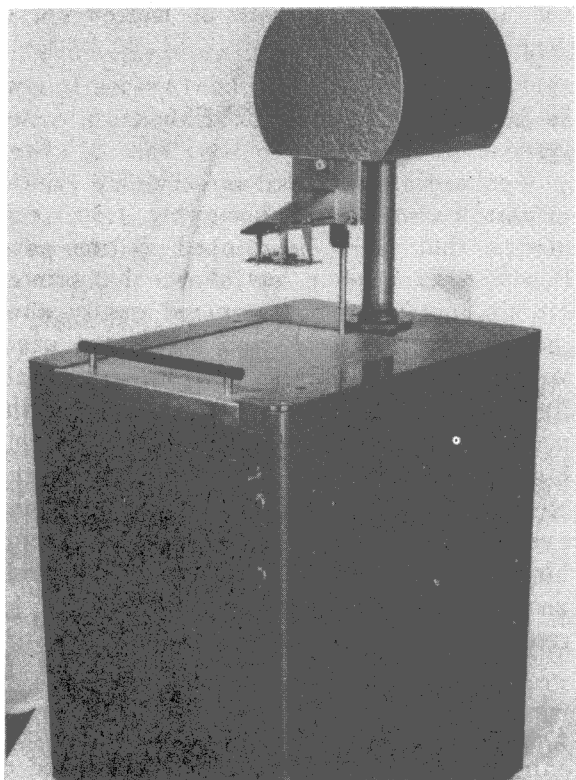
Tools: A small pencil-type soldering iron 35 to 50 watts; a pair of diagonal pliers; a pair of long-nose pliers, a soldering aid or scribe; a small firm bristly brush and a thin blade knife.

Materials: Rosin-core low temperature solder; cleaning alcohol, Krylon acrylic spray and tinned hook-up wire.

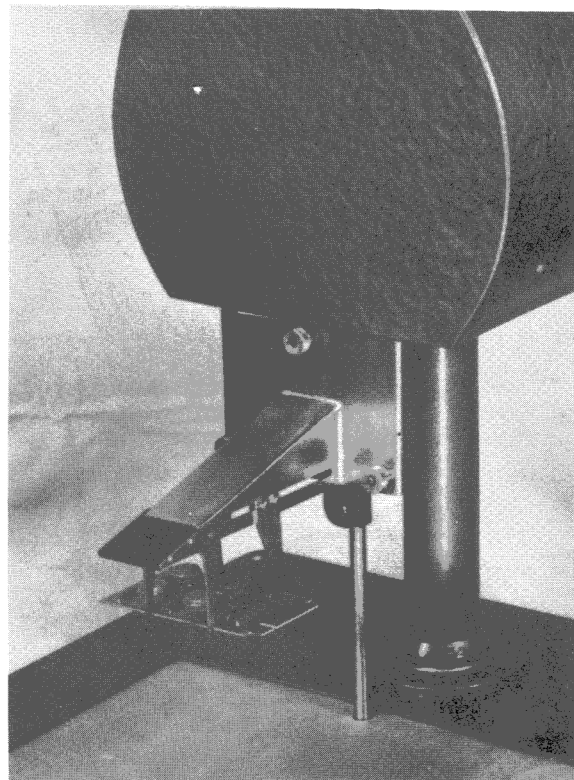
You can use wiring layout drawings and schematic diagrams to analyze printed circuit boards just as you use them to analyze conventional circuits. These drawings are provided in the manufacturer's service data. As a rule, you can analyze printed circuits without removing printed circuit boards from the chassis.

For convenience of mounting, replacements should be made with duplicate parts. You can make resistance or continuity measurements of coils, resistors, and some capacitors from the component side of the board. You can make voltage measurements on either side of the board.

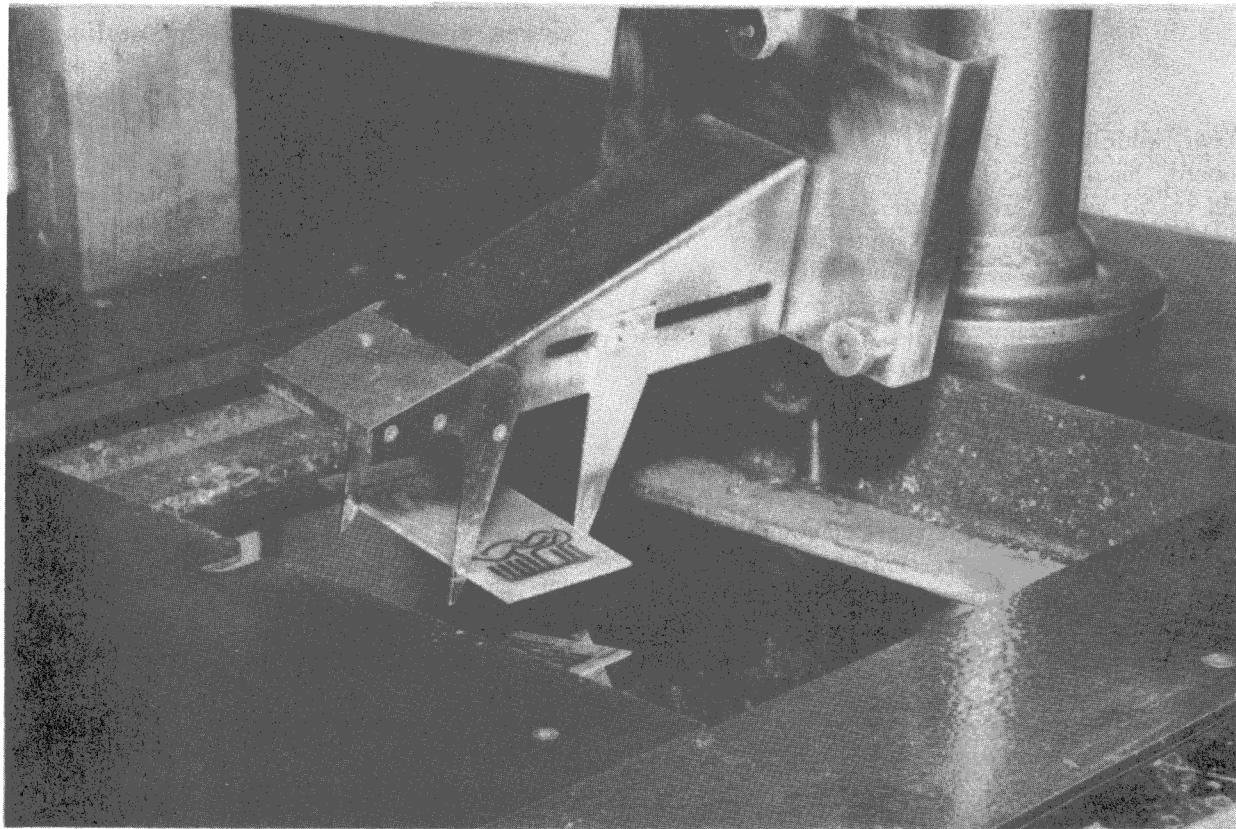
Correct reading data is generally supplied by the manufacturer and is frequently accompanied by specific data for the repair of the printed circuit board. Since tube socket terminals are often more accessible on printed circuit boards than on conventional circuits, measurements are often easier to make.



(a)



(b)



(c)

Fig. 39-19

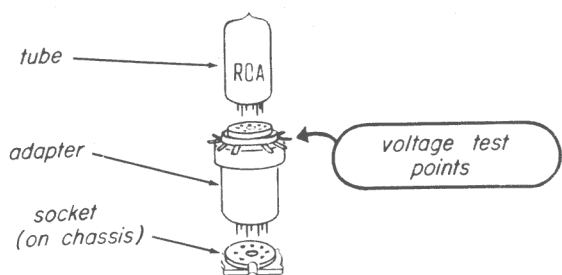


Fig. 39-20

If you can't make voltage measurements readily, you can check voltages at points common to tube socket connections; or you can use a tube-socket voltage test adapter. If you use an adapter, it should be of the longer type —preferably $1\frac{1}{4}$ inch long, since some of the sockets are recessed to a depth of $\frac{7}{8}$ inch. See Fig. 39-20.

The circuit is actually easier to trace out in many cases, partly because the printed circuit methods divide the apparatus into specific sections, making it easier to localize the trouble. For example, television receiver circuits are usually made up of more than one circuit board, because it is simpler and less costly to make such circuits in separate sections. Furthermore, a basic circuit section can be used in dif-

ferent models that vary considerable in other respects.

The *in-circuit capacitor tester*, illustrated in Fig. 39-21, can be used to advantage for troubleshooting printed circuits. The tester will locate most shorted, leaky, or open capacitors even though these parts are connected to the circuit. However, if a capacitor is shunted by a resistance of 5 ohms or less, it is difficult to check. Some in-circuit testers cannot be used for an accurate check of capacitors shunted by a high frequency coil (a coil with only a few turns). An oscilloscope can also be used with printed circuits just as it is used with conventional circuits.

The soldering iron used when working on printed circuit boards should have a wattage rating not greater than 50 watts, since excessive heat can readily damage a printed circuit board.

Replacement of Parts. When the faulty component has been located, the matter of replacement must be undertaken. As a rule, it is not necessary to remove the entire circuit board from the chassis.

Only if there is extensive damage to the printed connecting strips or a breakage of the board itself, should you replace the complete board. When you remove and replace components, take all possible precautions to prevent damage to the connecting strips bonded to the board and damage to the board itself. When you solder, do not permit excessive heat to damage the board. Also remember that excess solder causes shorts.

To replace capacitors or resistors on printed circuit boards use the following procedure:

1. Heat the solder joint on the printed side and draw the lead through (careful use of a knife will help loosen clinched leads).
2. Heat and clean out the lead hole with the scribe or soldering aid.
3. Form leads on new component and insert leads properly into the printed wiring.



Fig. 39-21

4. Re-solder leads using rosin-core solder, and remove excess flux with alcohol. Clip excess lead wire.

In instances when access to the underside of the board is difficult, components can be removed from the topside of the board as follows:

1. Cut the component being removed in half with a pair of diagonal cutting pliers (Fig. 39-22a).

2. Remove the body of the component from the connecting wires leaving as much wire as possible for connecting purposes (Fig. 39-22b).

3. In order to provide connecting points for the replacement component, cut the wire remaining so that the leads that remain to receive the replacement part are from 1/4-inch to 5/16-inch long. Form connecting loops as shown in Fig. 39-22c.

4. Run the lead of the replacement component through the loops of wire and bend the component leads to form a good connection. Solder the connection and cut off excess wire from the component leads.

If the printed circuit board has been removed from the receiver chassis for reasons other than component replacement, the parts may be removed simply by applying heat to the point on the connecting strip where the leads come through the board, bending

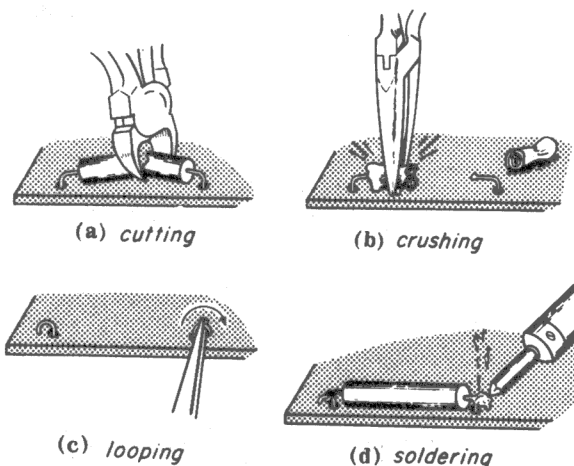


Fig. 39-22

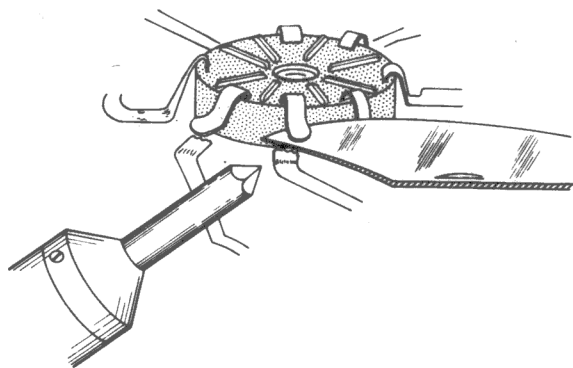


Fig. 39-23

the leads upright with a soldering aid, and lifting the part from the board. In the process of removing the solder, caution must be taken to prevent excessive heating. Use a small wire brush if necessary to quickly brush away the excessive solder from the connection. Do not leave the soldering iron on the connection when brushing away the solder. Melt the solder, remove the iron and quickly brush away the solder. It may require more than one heating and brushing process to completely remove the solder. The new part can then be mounted and secured in the original manner in place of the part that has been removed. Remove flush type tube sockets by melting and brushing the solder from the lugs and carefully lifting the lugs off the printed conductors to which they are attached (Fig. 39-23).

All printed circuit can be visually inspected. A magnifying glass will help to detect hairline cracks. If the circuit is one in which fairly high voltages occur, you may be able to find a break in the circuit by visual inspection in complete darkness with the apparatus on. Sometimes, a break will show up as a slight purple glow known as *corona*. High voltage discharge between adjacent conductors may also appear as *corona*. Locating the defect may prevent an actual breakdown of the board insulation.

Figure 39-24 illustrates the method usually employed to remove the elevated type of tube socket from a printed circuit board. Replacement of the sockets are simple since they are simply keyed in place and soldered.

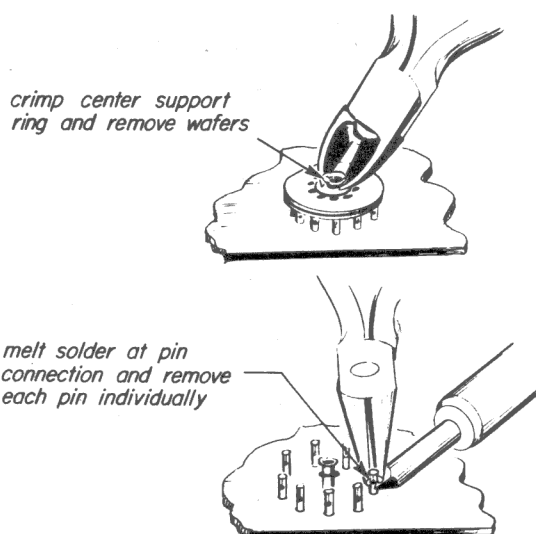


Fig. 39-24

The repair of break in copper wire is shown in Fig. 39-25 and is made as follows:

1. Cut a piece of hook-up wire about one-half inch longer than the break.
2. Tin both ends of the wire.
3. Center the wire over the break along the conductor and hold it with the soldering aid.
4. Heat both ends of the wire and hold it with the soldering aid until the solder sets. Then clean wire with alcohol.

Note: A small break in the continuity of printed wiring can normally be found easily by a visual examination of the circuitry. The use of a magnifying glass can be of help in locating small "hair line" breaks in printed wiring. Very small breaks can be repaired by carefully flowing solder over the break.

If the printed circuit wiring becomes raised from the board, clip off the raised section and replace it with a section of standard hook-up wire, secure each end at mechanically-sturdy tie points.

To remove i-f transformers and coils, apply just enough heat to the terminals to melt the solder so that the terminals can

be pushed away from the connecting strips. In some instances it may be necessary to carefully brush away excess solder from the terminals first. Installation of these parts is relatively simple since they normally snap into position and can be easily soldered.

If it is necessary to remove a printed circuit board, proceed with caution. A typical removal procedure is shown in Fig. 39-26. Do not attempt to force the board from each of its mounting lances; excessive flexing of the board may cause damage. Disconnect the wires where necessary from the printed circuit board. Cut and leave 1/8 inch of the color-coded wire covering to provide references facilitating the reconnection of the wires. These are wire-wrap connections, and once a wire-wrap connection is disconnected it should be reconnected by soldering. No attempts should be made to wrap the wire on the terminal once it has been removed.

To remove the board, apply heat to the point where the board eyelet is soldered to the lance and at the same time apply pressure to the back of the board through the hole in the chassis. When the solder melts, push the board out so that the lance is flush

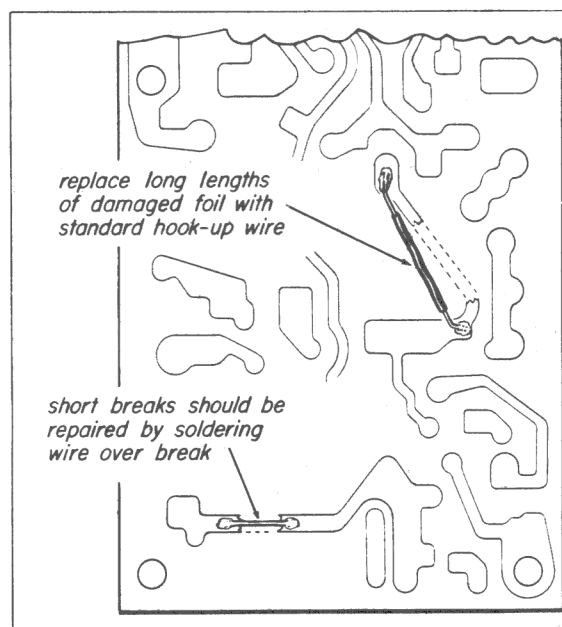


Fig. 39-25

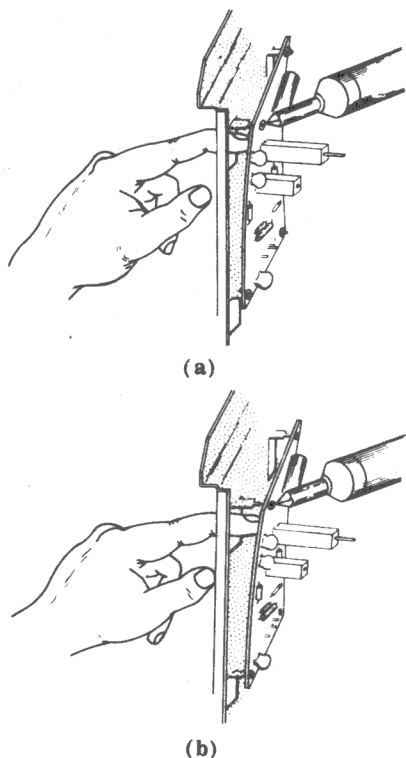


Fig. 39-26

with the eyelet in the board and let cool. Do not attempt to clear lance from eyelet at this point. Continue around the perimeter of the board systematically, doing the same at each lance. After each lance has been made flush with the eyelet, use the same technique to remove each lance completely from its eyelet, freeing the board from the chassis.

Repairing Modules. The repair of Erie P.A.C. modules is a fairly simple matter, since each component is soldered into its clips, and generally will be accessible. After the faulty component has been located, it can be unsoldered and replaced. A wedging tool can be a great help in this process.

It is not worthwhile to take the time and trouble to bypass an internally defective component in a couplate, even if this is possible. A far better job can be done by replacing the entire couplate.

On the other hand, it is better to try to repair Tinkertoy module circuits. The modules sometimes contain quite a number of parts. Replacement of the entire stack for one part might be unduly expensive. Before

you remove a wafer from a module, remove the module from the printed circuit board, after carefully marking the position of the module. The removal is done by the heating and brushing of solder and the gradual removal of the riser wires. After the module has been removed, remove all riser wires on one side of the stack as far down as the affected wafer carefully heating the risers and peeling them off (Fig. 39-27). The wafer to be removed is then gradually loosened from the remaining riser wires and slipped from the stack. A new wafer can be soldered into place; the riser wires can be restored to their original position; and the module can be reinstalled on the printed circuit board. In this process, the amount of heat used is not as critical as in the printed wiring boards, since the conductors on the wafers are fused to the ceramic at high temperatures, and are not as easily damaged. If a replacement wafer is not immediately available, repairs can also be made by simply removing the defective wafer restoring the riser wires with one empty space in the stack and then attaching a standard replacement component to the proper riser wires. Although this does not make a very good looking repair job, it may be a necessary emergency measure.

39-4. MAKING YOUR OWN PRINTED CIRCUITS

When printed circuitry is to be made in the shop or laboratory, the first problem to

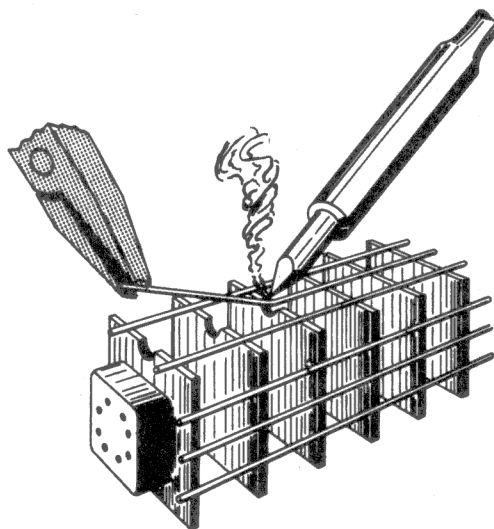


Fig. 39-27

be met is naturally the design of the circuit. Let's suppose that you wanted to design a printed circuit for the small remote control superregenerative receiver shown in Fig. 39-28. First, determine the approximate total size of the circuit. The size may have to be changed later. Next, determine the shape by arranging on a piece of cardboard the major components to be used and indicating the locations and terminals of tube sockets, coils, and relays. Interconnect the terminals of the components so that a minimum of crossovers occurs. One way to avoid crossovers is to take advantage of the spaces between the component leads. For example, when a capacitor is used, some separation will exist between the holes for the leads. This separation can be used for passing through other leads. The circuit design shown in Fig. 39-29 requires two crossovers, one for the filament connection and one for a relay terminal. One of these could have been avoided by selecting a different relative position for the relay. However, other considerations made its present location more desirable. The crossovers can be made with insulated pieces of wire on the component side of the board.

When you design a printed circuit, you may have to take into account some or all of the following considerations (not necessarily in the example shown here):

1. Current-carrying capacity of the copper foil conductors.

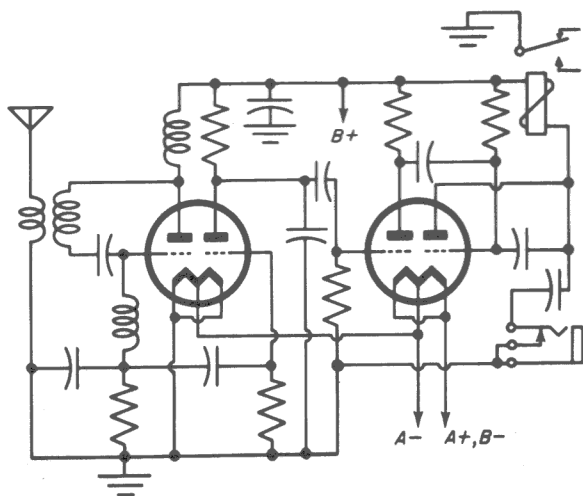


Fig. 39-28



Fig. 39-29

2. Insulation resistance of the plastic base.

3. The nature and spacing of adjacent conductors so that undesirable coupling effects and possible bridging by the solder will be avoided.

4. Possible unwanted stray capacitance produced by the conductors and the components.

5. Heat produced by the tubes that might melt the wax off capacitors or otherwise injure the components or circuit board.

6. On a-c circuit boards, no a-c current should be carried through grounding points for grid returns, etc. You may have to find grounding points for high frequency circuits.

After you have designed your circuit, the next step is to prepare the circuit board. There are several ways in which you can prepare printed circuit boards. Some of these ways resemble methods used in large-scale production. However, none of the methods described in this booklet requires special equipment or tools that are not easily available.

One method is shown in Fig. 39-30. The circuit has been carefully laid out, and the layout is used as a guide for the final circuit. In order to draw a final circuit, it is useful to have a complete set of components — a compass, a ruling pen, India ink, a straight edge and a triangle, as shown in Fig. 39-29. The conductors are drawn directly on 0.003 inch brass shim-

stock. (Shimstock means a very thin sheet of metal.) The cutout conductors are glued to the plastic base with one of the all-purpose glues now available from several manufacturers. The next step is to drill holes in the base to provide for component leads. A small drill, generally a No. 52 drill in a high-speed hand grinder, will do the easiest job. Eyelets should be inserted in the holes if the components are going to be experimented with and soldered and unsoldered several times. Aluminum or tin foil will do the job if solder contact need not be depended on. The eyelet must be pressed well enough to provide adequate electrical contact. Silver paint can be used to improve this contact between the eyelets and the foil after the components have been soldered in place.

Another method, which is quick but not as durable, is shown in Fig. 39-31. The base board is provided with the eyelets in the desired places and the conductors are painted on with silver paint. It is very likely that, after soldering, another layer of silver paint must be added to the connections of the conductive lines with the eyelets. In general, silver-paint conductors, unless cured and fused to the backing are not very durable and can easily be damaged by mechanical means or excessive current.

A method similar to the method mentioned above uses a laminated copper covered board. The desired conductor pattern can be cut out with a very sharp knife. Using this

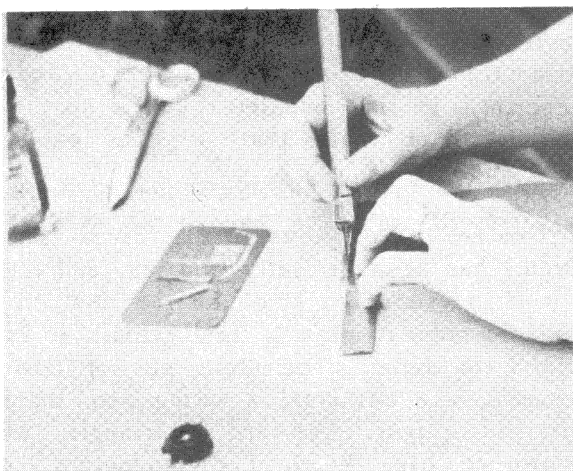


Fig. 39-30

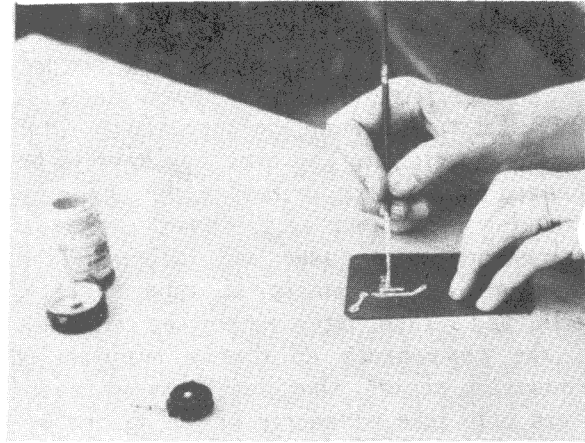
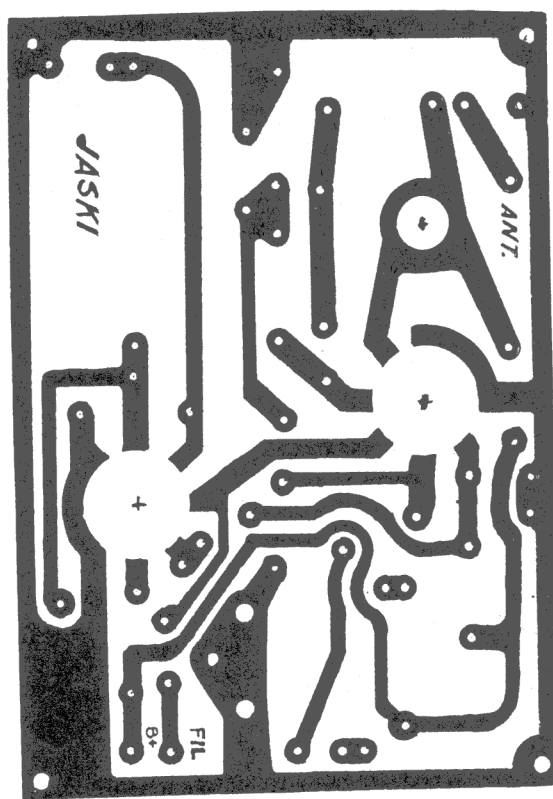


Fig. 39-31

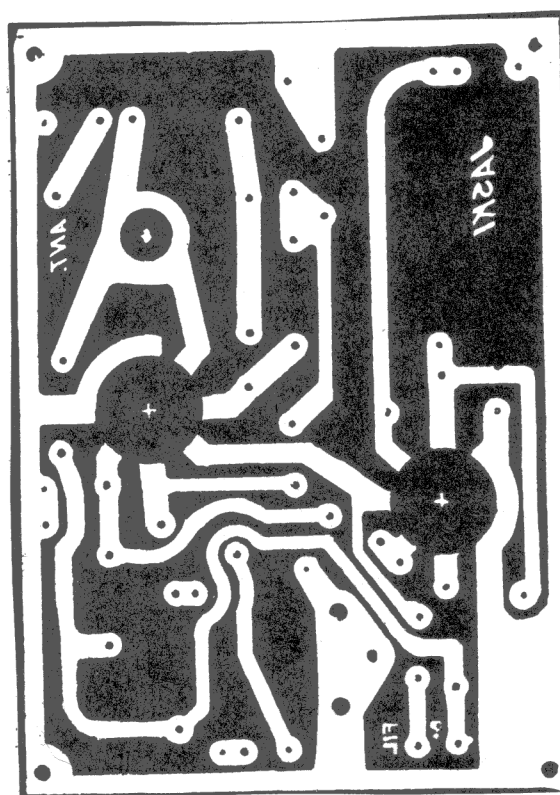
method, it is best to keep the conductors in straight lines. The undesired portions of the copper can then be peeled off by wedging the point of the knife under the copper and simultaneously heating the area with a very hot soldering iron. After the unwanted copper has been peeled off, a solvent can be used to remove the undesired bonding material.

If you cannot buy laminate without difficulty, you can make it. Although it will not be as good as the standard laminates, it will be good enough for many experimental purposes. It can be etched, cut, and soldered. It is made by gluing brass shimstock (with one of the all-purpose glues) to any phenolic or other thermosetting base. It is easier to use glue rather than a commercial bond for home-made laminate. The commercially used bond for laminating copper foil to phenolic requires high pressure and temperatures of 150 degrees. Temperature and pressure are somewhat critical to obtain a good bond.

The method for making individual printed circuits that is most nearly like production procedures is the method that produces an etched board. Several manufacturers produce kits with all the materials necessary to produce printed circuit boards of professional quality. But you don't have to buy a kit. The laminate is sold by the square foot at larger radio-supply houses, and the chemicals needed are also readily available. By assembling your own raw materials instead of using a kit, you can avoid limiting the size of your circuit board to the size of the board



(a)



(b)

Fig. 39-32

available in the kit. You can use one of three types of resist. One type is resist paint. You can use bituminous resist paint, a good quality acid-proof paint, or even an insulating varnish if the etching process is not too prolonged. You can paint the circuit on the laminate with a small brush.

Another type of resist is tape resist. Special narrow tape is sold for this purpose. Another narrow tape, which will work if carefully used, is the acetate tape sold in art stores for the production of illustrative charts. It is sold in widths ranging from 1/32-inch up to 3/8-inch. However, plastic electrical tape, cut into narrow strips with a utility knife and straight edge, can be used. Apply the tape wherever conductor lines are desired. The tape must be well pressed down, particularly in corners where tape overlaps; otherwise the etchant will undercut the tape and the board will have open conductors.

A third method of applying resist is the photographic method. The method and materials used are the same as those used in large scale production. The circuit is de-

signed and drawn carefully on a piece of drawing paper in a larger size than it will have in its final form, as shown in Fig. 39-30. Then it is photographed. Figure 39-30 shows the drawing of the circuit on the drafting board. Figure 39-32a shows the positive reduced to exact size, and Fig. 39-32b shows the paper negative produced from the positive. Film negatives can be made, but it is more difficult to obtain the required contrast with them. The paper negative can be stored for use later on. The resist that is used in this process, can be bought from Eastman Kodak, but this resist is more expensive than home-made resist. A small amount of very readily available chemicals will produce a professional quality glue-top resist at low cost. The chemicals are fish glue, (or *LePage's* photo-engravers glue), distilled water, and ammonium bichromate. To one part of fish glue, add two parts of distilled water, and for each cup of liquid, add two tablespoons of bichromate. Stir this well, filter, and pour it into a brown bottle. In 24 hours, the resist will have obtained its greatest sensitivity to light. Store the resist out of strong light and in a cool place; refrigeration is not necessary.

In addition to the resist, you need an etchant, or etching solution. The most commonly used is ferric chloride dissolved in water. Chunks of ferric chloride can be purchased from industrial-chemical supply houses. It is possible to etch with diluted nitric acid, or a combination of nitric and sulfuric acid, but these etchants are far more dangerous to handle.

The next step in the process of making an etched printed circuit is the preparation of the circuit board. The circuit board first should have a very thorough cleansing. Then the board should be washed with etchant. This process called pre-etching, makes the copper rough so the resist will stay on it. After a few minutes of pre-etching, the board should be very thoroughly washed. No etchant should remain on the board. Next, the resist solution should be applied to the board, preferably by flowing it on, then whirling the board in some manner to get even distribution. This entire operation of applying the resist and whirling or drying must be carried out in the dark or in subdued light.

The next step is to place the paper negative on the board and expose the board to a number 2 photoflood at a distance of approximately 3 feet for about six to ten minutes. The time of exposure depends somewhat upon the humidity of the place in which you are working.

Next remove the negative and wash the exposed board in water. The circuit pattern will show. If you want to make the pattern stand out, you can dye it with ordinary food colors. Next, immerse the board for about 30

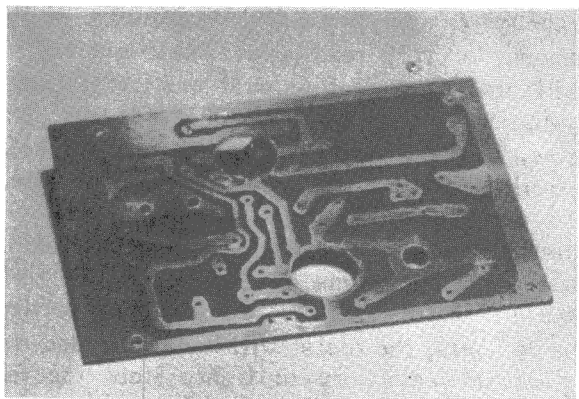


Fig. 39-33

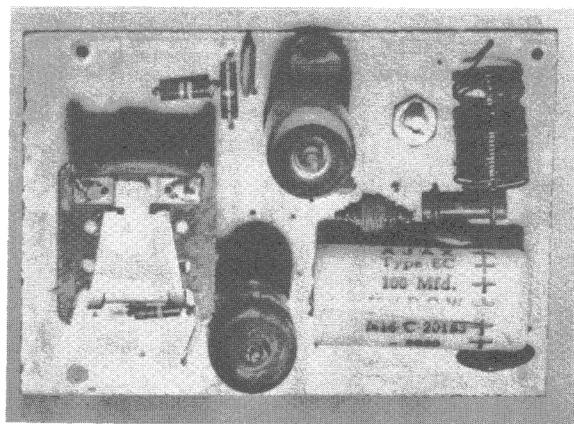


Fig. 39-34

seconds in a 5-percent solution chromic acid in order to harden the resist. Dry the board under a photoflood lamp, an infrared lamp, or in a domestic oven at a temperature lower than 200 degrees; higher temperatures would damage the image as well as the board. Then immerse the circuit board in the etchant. Etching in a stationary bath may take 45 minutes or more. After etching, the board should be thoroughly washed and dried. If a few hairline cracks appear, they can easily be bridged with a little solder. It would be well to check continuity of the conductors with an ohmmeter before the components are installed. The remaining resist should be removed with fine steel wool and the board punched for component leads or eyelets. Figure 39-33 shows the completed etched board. Figure 39-34 shows the components installed on the board.

Plating can be another useful method for the preparation of circuit boards in the laboratory, but the methods used in production must be modified. The circuit conductors can be painted on a base with all the conductors connected to a silver-painted edge. Then, after plating, the edge can be sanded off to remove the interconnection. Great care must be taken to have the board absolutely clean, or the plating will not bond. The silver conducting paint must be applied as thin as possible, and the plating bath, whether it be cyanide or acid must be accurately controlled for temperature. Too hot a bath will loosen the bond; too cold a bath will give porous copper. The voltage value is also extremely important.

Table A gives data about plating solutions, temperatures and voltage values.

What is needed for plating is an acid-proof bath (container) preferably glass, some clip-leads to connect the electrodes to a voltage source, a d-c source, and a solution of diluted sulfuric acid with copper sulfate and Rochelle salts. The bath must be placed so that it can readily be heated to about 140 degrees, and the best way to do this is to set the bath inside another pan that constantly receives hot water running from a faucet. Most domestic water heaters supply 140 degrees water.

The board to be plated is connected to the anode (+) of the source; the cathode (-) will be a piece of pure copper. It is best to have a plate for cathode about the same size as the printed circuit board to be plated, so that the copper particles will distribute evenly. With this setup, you can plate the required thickness in the time shown in Table A.

If you use printed capacitors, use the following formula to calculate their capacitances:

$$A = \frac{Cd}{0.224 K (N-1)}$$

where

A = area of one printed area in square inches

C = capacity of the entire capacitor in micromicro-farads

K = the dielectric constant

d = thickness of the dielectric in inches

N = the total number of printed areas

The dielectric constant can be obtained from the supplier of the base material. These are representative of most printed circuit laminates.

If you use spiral printed conductors, their inductances may be found from the following formula:

$$L = 0.00194 dn^2 u$$

where

d = outside diameter of the spiral

n = number of turns

u = permeability of the core and other material around the coil

For nonmagnetic materials, $u = 1$

TABLE A - ELECTROPLATING DATA

Metal	Type of Solution	Principal Ingredients	Temperature	Volts	Time to Deposit 0.001 inch (minutes)
Copper	Acid	CuSO ₄ , 5H ₂ O, H ₂ SO ₄	75 to 120	1 to 2	35
	Rochelle	CuCN, NaCN, Na ₂ CO ₃ plus Rochelle	140 to 160	2 to 3	45
	Cyanide	CuCN, NaCN, Na ₂ CO ₃	75 to 100	1.5 to 3	90
Silver	Cyanide	AgCN, KCN, K ₂ CO ₃ , CS ₂ (or Na in place of K)	80	1	
Gold	Cyanide	KAu(CN) ₂ , K ₂ CO ₃ , KCN	120 to 160	2 to 6	